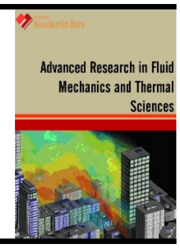
Penerbit Akademia Baru

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences

Journal homepage: www.akademiabaru.com/arfmts.html
ISSN: 2289-7879



Assessment of Turbulence Modelling for Numerical Simulations into Pico Hydro Turbine

Open Access

Dendy Adanta[1,*], Budiarso[1], Warjito[1], Ahmad Indra Siswantara[1]

[1] Department of Mechanical Engineering, Faculty of Engineering, Universitas Indonesia, 16424, Depok, Indonesia

ARTICLE INFO

Article history:
Received 28 February 2018
Received in revised form 25 April 2018
Accepted 9 May 2018
Available online 14 June 2018

ABSTRACT

Currently the computational fluids dynamics (CFD) method is becoming an important subject of research in engineering, and pico hydro seems to be of particular interest. To increase accuracy using the CFD method, the assumptions made should be close to the actual conditions. However, there has been no comprehensive study that explains the characteristics and turbulent models that are considered suitable for use in in the pico hydro turbine. This study aims to explain flow characteristics to determine whether turbulent flow would occur and recommends a turbulent model that may be applied to a pico hydro turbine. To achieve the objectives of the study, several methods are used, including asymptotic invariance (Reynolds number analysis), local invariance, theoretical analysis and a literature study. This study found that the flow profile that occurs is irregular; the Reynolds number flow is 420,972, within the turbulent flow category; vorticity occurs with the prediction using isotropic assumptions; flow dissipation occurs; and is continuous because turbulent kinetic energy is supplied from the main flow. Thus, the category of water flow in a pico hydro turbine with power potential 1 kW is turbulent. The literature study reveals that the prediction of turbulent flow in the pico hydro turbine can be realized by three models: standard k-ε is recommended for the overshot waterwheel, RNG k-ε is recommended for the undershot waterwheel and cross-flow turbine, SST k-ω is recommended for propeller or openflume, Pelton, breastshot waterwheel and Turgo turbines. However, these recommendations do not constitute a final conclusion because a good turbulent model is based on actual conditions.

Keywords:
Pico hydro, CFD, turbulent flow, turbulent model.

1. Introduction

The three main methods used in this study of the pico hydro turbine are a literature study, the experimental method and the computational fluid dynamics (CFD) method. The literature study is used to establish the state of research in pico hydro turbines [1]. The experimental method is used for determining turbine performance and developing the basic theory of turbine design [2]. Finally, the CFD method is used to determine turbine performance, which is then used for optimization [3].

[*] *Corresponding author.*
E-mail address: dendyadanta@ymail.com (Dendy Adanta)

There have been many studies that used the methods above of pico hydro turbines; these studies were intended to determine which components or turbines are suitable in certain conditions. Williamson and Simpson [4] conducted a feasibility study of independent power plants for remote areas and concluded that the pico hydro turbines would be suitable for those environments. Williamson and Simpson [4] suggested diameter sizes of penstock for use with maximum 10% head loss. A feasibility study was performed by Lahimer *et al.,* [5], who concluded that pico hydro is the most effective means of supplying household electricity needs due to lower investment and operational costs than other alternatives. To discover which turbine types were suitable under low head conditions (<3.5 m) Williamson *et al.,* [1] analysed 13 types of pico hydro turbines using quantitative and qualitative methods. Williamson *et al.,* [1] concluded that a propeller turbine with a draft tube and a single-jet Turgo turbine was most suitable for remote areas.

Researchers have also examined the electrical systems associated with pico hydro. Haidar *et al.,* [6] reviewed the typical generator and electrical system used with pico hydro. From the experimental results, Haidar *et al.,* [6] recommend a DC current due to its low voltage and the ampere generated. Gladstone *et al.,* [7] studied the most appropriate electrical system to use in pico hydro systems. The result of the study determined that the battery is suitable because the generated power is low and fluctuating, so that it cannot be directly utilized; it may shorten the lifetime of electronic equipment. Ridzuan *et al.,* [8] analysed the piping system without a forebay to determine the effect of turbine power and found that the DC electrical system was suitable for pico hydro.

The previous studies have not reviewed the turbulent models used in each type of turbine. Turbulent models are used to predict the flow pattern that will occur when the turbine is in use. Accurate turbulent models will produce CFD results that can represent actual conditions. Its developing technology makes the CFD method widely used to facilitate visualization of physical phenomena not conducted using other methods, is no exception on pico hydro turbine [9]. The precise result of the CFD method on the pico hydro is turbulent model. The turbulent model is assumed because it provides computational results such as torque, power, and efficiency [10]. However, studies using turbulent models of pico hydro were based only on assumptions such as k-ε, turbulent models used because they use only two (simple) equations and have fairly low computational power. The SST k-ω turbulent model is used because it can represent turbulent flow near the wall and so on.

This study aims to determine the nature of the turbulent flow produced and recommends a turbulent model that may be applied to each type of pico hydro turbine. This work is a continuation of previous work [11]. However, the previous work did not clearly demonstrate whether the flow occurring is turbulent or laminar and only reviewed three types of turbine.

2. Methodology

2.1 The basic equation on turbulent flow

There are many basic equations that are used to describe turbulent flow. These include the Reynolds number (R_e), boundary layer thickness turbulent (δ_{turb}), mixing length, turbulent kinetic energy, turbulent rotational speed, and dissipation rate. Determination of Reynolds number (R_e) using Eq. 1 [12]:

$$R_e = \frac{\rho U D}{\mu} = \frac{UD}{v} \tag{1}$$

The boundary layer thickness turbulent is a stream line formed by flow through the flow field. The boundary layer thickness turbulent (δ_{turb}) using Eq. 2 [13]:

$$\delta_{turb} = 0{,}37 {R_e}^{-1/6} \mathrm{X} \tag{2}$$

The mixing length (l_m is a random displacement process of the fluid particles over a certain distance [14]. From the experimental results, the l_m is determined by Eq. 3 or 4 [15]:

$$l_m = K.y = \delta \tag{3}$$

$$l_m = \delta \tag{4}$$

Eq. 3 for $y < \delta$ and Eq. 4 to $y \geq \delta$.

Fluctuations in the fluid velocity vector cause fluid shear stress between the particles; this is represented by Eq. 5 [14]:

$$\tau_{turb} = \rho {l_m}^2 \left(\frac{d\bar{u}}{dy}\right)^2 \tag{5}$$

The turbulent kinetic energy (k) is the energy contained by the magnitude of a flow vortex [15]. Eq. 6 representation of k [15]:

$$k = (\bar{u}^2 + \bar{v}^2 + \bar{w}^2)/2 \tag{6}$$

To facilitate the analysis of turbulent flow, the most commonly used assumption is isotropic (the velocity of the three axes is considered to be the same) [15], [16]. So Eq. 6 becomes:

$$k = \frac{3}{2}(\bar{u}'^2) \tag{7}$$

Tennekes and Lumley [16] observe that the rotational velocity eddy (u) divided by average velocity ($\overline{U}$) is proportional to the boundary layer thickness (δ) divided by the characteristic length of the flow field (X) or $\delta/X \sim u/U$.

The dissipation rate (ε) is the energy transfer from high to low eddies [16]. ε on the turbulent flow is dependent on viscosity and the velocity gradient *(shear)*. Eq. 8 representation of ε [17]:

$$\varepsilon = 15v\overline{(\partial u_1/\partial x_1)^2} \tag{8}$$

2.2 Kolmogorov's Theory

There are two basic hypotheses of Kolmogorov's theory: isotropic and anisotropic assumptions. The difference between these two assumptions is in the two boundary layer thicknesses formed: with an isotropic of $l \ll l_0$, velocity function becomes $u'^2 = v'^2 = w'^2$, with an anisotropic of $l < l_{EI}$, the value of the boundary layer thickness is $l_{EI} \approx l_0/6$ [17]. Kolmogorov in calculating turbulent flow introduced the Reynolds number to Kolmogorov's scale ($R_{e\eta}$) [17]. There are three scales: length scale, velocity scale and time scale [16].
length scale (η):

$$\eta = \left(\frac{v^3}{\varepsilon}\right)^{1/4} \tag{9}$$

velocity scale (u_η):

$$u_\eta = (\varepsilon v)^{\frac{1}{4}} \tag{10}$$

time scale (τ_η):

$$\tau_\eta = (v/\varepsilon)^{\frac{1}{3}} \tag{11}$$

2.2 Taylor Microscale

The Taylor microscale (λ) is the definition of turbulent values in sub-inertia [17]. From the experimental result, λ is function of velocity: $\overline{(\partial u_1/\partial x_1)^2} \equiv \overline{u_1}^2/\lambda^2 = u'^2/\lambda^2$. So, to determine dissipation rate from the Taylor microscale [17]:

$$\varepsilon = 15 v u'^2/\lambda^2 \tag{12}$$

$k = (1/2)(u'^2 + v'^2 + w'^2)$, $u'^2 = (\frac{2}{3})k$, so $\varepsilon = 10vk/\lambda^2$, and then $\lambda \approx (10vk/\varepsilon)^{1/2}$.
The Reynolds number Taylor scale (R_λ) is a function of the length scale, and the length scale is a function of the velocity scale, from Taylor's hypothesis that the R_λ is defined from Reynolds number [17]:

$$R_\lambda = u'\frac{\lambda}{v} = \left(\frac{20}{3}\mathrm{R_e}\right)^{1/2} \tag{13}$$

2.3 The Energy Transfer Rate and Wavenumber (κ)

Some CFD software uses a η to determine the wavenumber (ηκ). Kolmogorov established a ratio of values for isotropic assumptions at a small scale. The previsions in reference are [17]: first, the magnitude of the initial eddy (l) should be smaller than the eddy at the end of energy production (l_{Ei}) or $l < l_{Ei}$. Second, the assumption for eddy quantity at the point of energy production ends (l_{Ei}) is $l_{Ei} \approx l_0/6$. And third, the eddy size at the beginning of the dissipation rate (l_{DI}) is 60 times the length scale ($l_{DI} \approx 60\eta$).

2.4 Spectrum Normalization

Spectral normalization is a way of knowing the analytical position of energy production, sub-range inertia and when the rate of dissipation is ended. The dimensional volume of the energy spectrum per acceleration by flow is $m^3 s^{-2}$ [17]. To get dimension volume per acceleration requires analysis of dimensions k, ε and κ. The dimensional analysis for $k = m^2 s^{-2}$, $\varepsilon = m^2 s^{-3}$, $\kappa = m^{-1}$ so that the energy spectrum is $E(\kappa) = \frac{k}{\kappa} = m^3 s^{-2}$ or $\varepsilon^{2/3}\kappa^{-5/3} = m^3 s^{-2}$, so [17]:

$$E(\kappa) = C\varepsilon^{2/3}\kappa^{-5/3} \tag{14}$$

C is Kolmogorov's constant (C = 1.5). In CFD software, to determine the overall energy spectrum, new functions have been added to the analysis: production range (f_L) and dissipation range (f_η):

Production range (f_L):

$$f_L = \left(\frac{\kappa l_0}{[(\kappa l_0)^2 + C_L]^{1/2}}\right)^{Po+5/3} \tag{15}$$

Dissipation range (f_η):

$$f_\eta = exp\left\{-\beta\left\{\left[\left[(\kappa\eta)^4 + {C_\eta}^2\right]\right]^{1/4} - C_\eta\right\}\right\} \tag{16}$$

Eq. 15 and 16 added to the left side of Eq. 14:

$$E(\kappa) = C\varepsilon^{2/3}\kappa^{-5/3} f_L f_\eta \tag{17}$$

The constants are determined from the previous study: $C_L \approx 6.78; C_\eta \approx 0.40; C = 1.5;\ P_0 = 2; \beta = 5.2$. This study only uses energy spectrum Kolmogorov's scale, which in the label x is ηκ and in the label y is $E(\kappa)/(\eta {u_n}^2)$.

2.4 Method

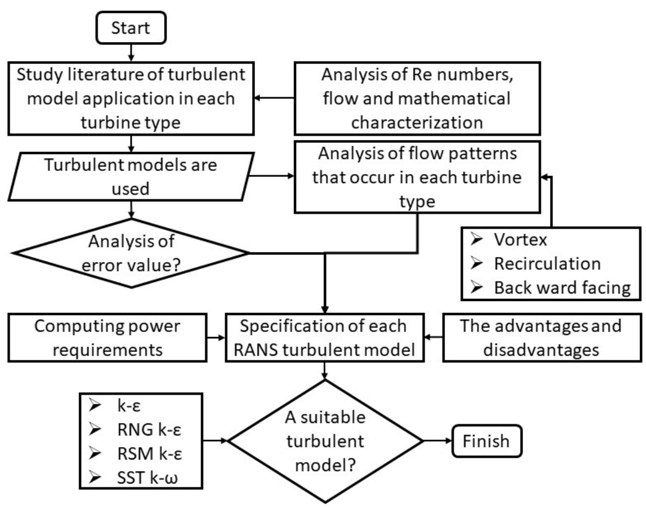


Fig. 1. Flow chart of study

This study uses two methods used: theoretical analysis and a literature study. The theoretical analysis involves three elements: asymptotic invariance, local invariance and mathematics (calculating the value of turbulent kinetic energy, the dissipation rate and the energy spectrum) [16]. The literature study is used to analyse the advantages and disadvantages of each turbulent model, such as their governing equations, their computing power requirements and their usability specifications. In addition, the study seeks to determine the use of the turbulent model in pico hydro turbines, then applies turbulent model based on error analysis (comparison of the simulation with experimental results). The activities undertaken in this study can be seen in Figure 1. The measurement of the flow velocity distribution was taken using a pitot tube connected to a pressure sensor and sent to a computer using DAQ (data acquisition). The length of the observation is 0.85

meters, the height of the first point (x = 0) is 0.34 meters and height of the second point (x = 0.85) is 0.36 meters.

3. Results and Discussions

3.1 Theoretical Analysis

The result of observation by the asymptotic invariance method found that the Reynolds number on the pico hydro turbine open channel is 460,972, which indicates a turbulent flow ($Re > 200{,}000$) [14]. The local invariance method found that the flow characteristics were steady and non-uniform. Steady means that the geometry of the passage does not change with time (spatial) [18]. Non-uniform means the properties of the stream change from point to point so as to make the shape of the flow change over time (temporal) [18].

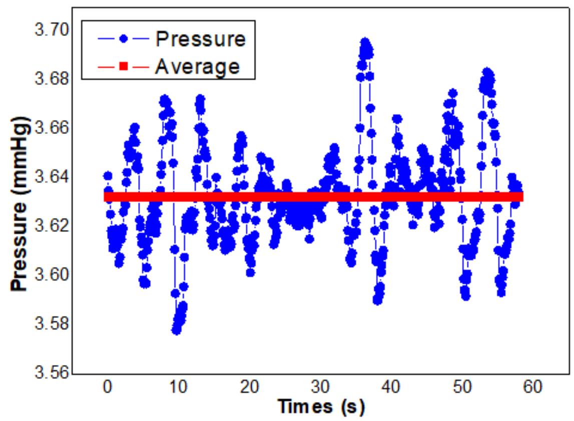


Fig. 2. The pressure dynamic fluctuation measurement at y=0.5 and x=0.5

Measurement of the pressure water dynamic from 0.05 to 0.34 cm flow height are categorized fluctuation. The measured dynamic pressure is converted to velocity so that velocity over time fluctuates and also indicates turbulent flow [12]-[13]. Visualization of the pressure dynamic fluctuations is represented in Figure 2. It is assumed that the flow velocity on the wall (y=0) is 0, so the velocity distribution from y=0 to y= 0.34 m is:

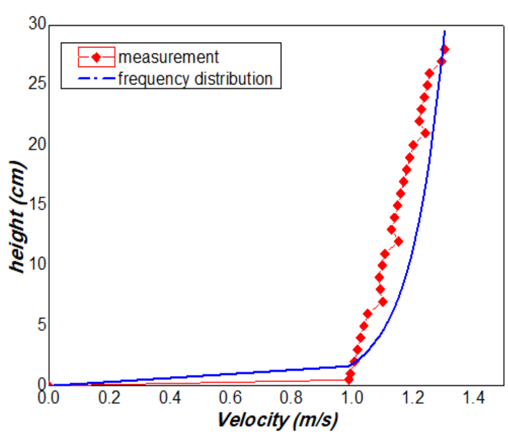


Fig. 3. The velocity distribution at x=0 by present work

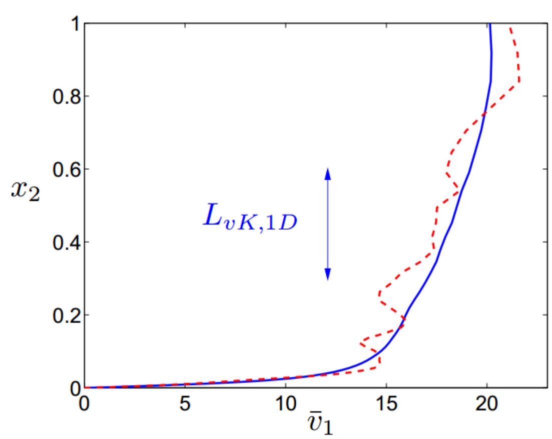


Fig. 4. The velocity distribution by Davidson's study

The observed results show the average velocity flow ($\bar{u}$) of 0.798 m/s on a height of ± 2 cm and with an average turbulent intensity (*TI*) of 3%. The exponential approach is used to facilitate the prediction of velocity profiles located within a certain height, and the indications of velocity distribution are similar when compared to Davidson's study [19]. The velocity profile in Figure 3 resembles a turbulent flow profile with an *n* value of 8.5 [14].

The determination of a turbulent model depends on the understanding of turbulent shear stress. The shear stress is caused by the random motion of particles; in this case, that motion is particle velocity (flow velocity). From the result of plotting farther from the wall, turbulent shear stress decreases because turbulent intensity (*TI*) increases. Based on measurement, the maximum turbulent shear stress of 0.712 N with an indication is turbulent flow [14].

Drag coefficient is a function of the roughness of the wall surface [16]. Thus, turbulent flow is also affected by this roughness. The level of turbulence flow can be known through an analysis of the drag coefficient. In Figure 5, the maximum drag coefficient of 0.041 is categorized as a full turbulent flow (see Heald's graph) [20].

The dissipation rate occurs when the mixing length (l_{Di}) is proportional to 60 times the length scale (η) ($l_{DI} \approx 60\eta$). The results of the calculations performed are similar to those of Kolmogorov and Taylor [17]. In the inertial subrange, the curve formed is linear because the energy cascade in the mean flow is conservative. This is because the total amount of energy dissipated per unit mass and time is ε and the spectral energy flux is equal to ε [16]. In addition, when viewed from the curve pattern, the curve shape in Figure 6 is similar to Lesieur's analysis in its low Prandtl number [21].

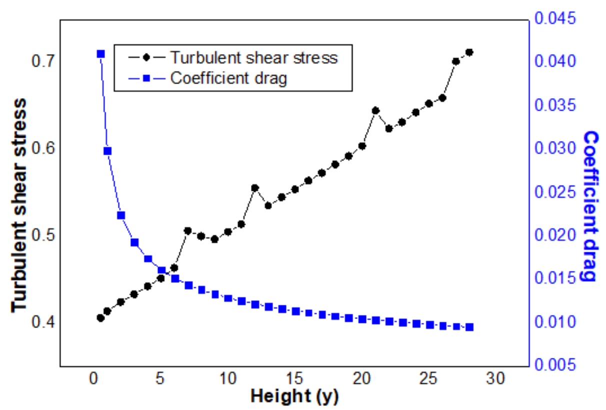


Fig. 5. The height flow relationship to turbulent shear stress and coefficient shear drag

Fig. 6. The graphic of energy spectrum using Kolmogorov's scale

3.2 The Literature Study

Before discussing the application of turbulent models that have been used in pico hydro turbines, the advantages and disadvantages of each turbulent model will first be discussed. Table 1 represents advantages and disadvantages of turbulent models.

Table 1
Comparison of Reynolds Average Navier-Stokes (RANS) turbulent models

Description	Advantages and disadvantages of turbulent model			
	k-ε	RNG k-ε	SST k-ω	RSM k-ε
Governing equation	2 governing equations (k - ε) [19]	2 governing equations (k - ε) + inverse Prandtl number (α) [10]	4 transport equations (k-ω) + area near wall (F1) and strain rate magnitude (F2) [22], [23]	7 transport equations (convection + Production + strain pressure + Diffusion (time and viscosity) + production buoyancy + dissipative) [19]
Usability specification	Good enough for general flow simulation [10]	Special for swirl flow analysis [10]	Validation turbulent model using rotating velocity flow [22], [23]	Suitable for complicated flow and have convection [19]
Power computing	Simple	General	Medium	High

The literature study included 14 studies concerning seven turbines (openflume, Pelton, Turgo, cross-flow, undershot, breastshot, and overshot turbine) that compare simulation results with experimental.

Table 2
Analysis of the application of the turbulent model and turbulent flow on pico hydro

Type of turbine	The vortex flow patterns that occur	Error simulation with experimental			
		k-ε	RNG k-ε	SST k-ω	RSM k-ε
Propeller	Secondary flow [24] and plug-hole vortex [25]	50% [26] and 10% [27]	-	0,2 - 2% [28] and 4% [29]	-
Pelton	Irrotational vortex [30]	Not good [31]	11% [27, 28]	3,5% [31] and 4,8% [33]	-
Turgo	Secondary flows [34]	17% [35]	-	Very Good [36]	-
Cross-flow	Flow recirculation [37]	22% [38]	4% [39]	-	28% [40]
Undershot	Eddies flow [41]	-	Recommended	-	-
Overshot	Free-surface vortex [42]	Recommended	-	-	-
Breastshot	Free-surface vortex [42]	-	5% [43]	-	-

3.4 Discussion
3.4.1 k-ε Standard

There are two basic reasons why k-ε is suitable for the overshot waterwheel. First, turbulent flow occurs only in the water channel. Second, when the water is in the bucket, the turbulent flow does not significantly affect the energy conversion process. This means that the turbulent flow is categorized as simple. The k-ε turbulent model is suitable means of obtaining results for use when computing power is a consideration.

3.4.2 Renormalization Group (RNG) k-ε

The RNG k-ε turbulent model is fairly simple but can produce more precise results than standard k-ε because the flow near the wall is analysed more thoroughly [44]. In addition, the development of this model is mostly done by the recirculation flow that occurs in the internal impeller cross-flow (turbine or fan) [37], [45], [46]. However, for simple cases this model is considered too diffuse if the value α (inverse Prandtl number) is not adjusted to the actual conditions [10].

After the water hits the blades of the undershot and cross-flow turbines, the stream flow expands, resulting in an increasing pressure that causes recirculation flow. The recirculation flow must be precisely identified because this flow can decrease the water power to be converted [47]. As seen in Table 1 and 2, it is reasonable that the RNG k-ε has an error value smaller than other models.

3.4.3 Shear Stress Transport (SST) k-ω

SST k-ω is a turbulent model developed by Menter [22], [23] that combines two equations: k-ε and k-ω. Validation of this model is done by comparing the velocity of backward facing flow with an error value 5% lower than baseline (BSL) k-ω model of 8% [19], [23]. This is because this model changes the adverse pressure gradient into eddy-viscosity [23].

The secondary flow occurs in the propeller, openflume or Kaplan turbines apparently because the pressure on the top of the blade is lower than the bottom, so a vortex occurs even though the water velocity toward the outlet side is still more dominant [24]. The plug-hole vortex is indicated due to a pressure decrease in the outlet side causing the water to be sucked into the draft tube. The plug-hole vortex is a representation of the head energy of water. On the Turgo turbine secondary flow also occurs because of an imbalance between pressure and centrifugal force in the wall due to the curved flow field [34, 48, 49]. The Pelton turbine creates an irrational vortex because the water from the jet follows the shape of a semi-circular bucket. This causes the water to have a tangential force so that water whirls near the wall of the bucket [33]. Secondary flow, a plug-hole vortex and an irrational vortex is turbulent flow which is influenced by scalar and vector of the flow velocity [50].

In propeller, Kaplan, Pelton and Turgo turbines, the literature study found that SST k-ω has an error value of under 5%, less than other models. The prediction of the turbulent flow in the Pelton turbine was carried out using this model. The Pelton turbine was chosen because it has a complex bucket geometry compared to the propeller and Turgo turbines and requires boundary conditions in two phases [33]. From the simulation result, SST k-ω can represent the actual condition of the Pelton turbine where the error value is 4.8% [33].

4. Conclusions

There are five characteristics of turbulent flow: irregularity, a large Reynolds number, vorticity, dissipation and continuum [16]. The results of the study determined that the flow profile that occurs is irregular (see Figure 3); the Reynolds number flow is 420,972, within the turbulent flow category; vorticity occurs with the prediction using isotropic assumptions; flow dissipation occur (see Figure 6); and is continuous because turbulent kinetic energy is supplied from main flow. Thus, the category of water flow in a pico hydro turbine with power potential 1 kW is turbulent flow.

From the literature study, the prediction of turbulent flow in a pico hydro turbine can be performed using three models: standard k-ε is recommended for the overshot waterwheel, RNG k-ε is recommended for the undershot waterwheel and cross-flow turbine, while SST k-ω is recommended for propeller or openflume, Pelton, breastshot waterwheel and Turgo turbines. However, these recommendations do not constitute a final conclusion because a good turbulent model is based on actual conditions.

Acknowledgement
This research was funded by a grant from the Directorate of Research and Community Service (DRPM) University of Indonesia, grant no. 1343/UN2.R3.1/HKP.05.00/2018

References

[1] Williamson, S. J., B. H. Stark, and J. D. Booker. "Low head pico hydro turbine selection using a multi-criteria analysis." *Renewable Energy* 61 (2014): 43-50.
[2] Nechleba, Miroslav. "Hydraulic turbines, their design and equipement." (1957).
[3] Janjua, AJAZ BASHIR, Muhammad Shahid Khalil, and Muhammad Saeed. "Blade profile optimization of kaplan turbine using cfd analysis." *Mehran University Research Journal of Engineering and Technology* 32, no. 4 (2013): 559-574.
[4] Williams, A. A., and R. Simpson. "Pico hydro–Reducing technical risks for rural electrification." *Renewable Energy* 34, no. 8 (2009): 1986-1991.
[5] Lahimer, A. A., M. A. Alghoul, Kamaruzzaman Sopian, Nowshad Amin, Nilofar Asim, and M. I. Fadhel. "Research and development aspects of pico-hydro power." *Renewable and Sustainable Energy Reviews* 16, no. 8 (2012): 5861-5878.
[6] Haidar, Ahmed MA, Mohd FM Senan, Abdulhakim Noman, and Taha Radman. "Utilization of pico hydro generation in domestic and commercial loads." *Renewable and Sustainable Energy Reviews* 16, no. 1 (2012): 518-524.
[7] Gladstone, Scott, Victoria Tersigni, Kevin Francfort, and Julie Ann Haldeman. "Implementing pico-hydropower sites in rural Rwanda." *Procedia Engineering* 78 (2014): 279-286.
[8] Ridzuan, Mohd Jamir Mohd, S. M. Hafis, K. Azduwin, K. M. Firdaus, and Zawawi Zarina. "Development of pico-hydro turbine for domestic use." *Appl Mech Mater* 695 (2014): 408-12.
[9] Warjito, Dendy Adanta, Budiarso, and Aji P. Prakoso. "The effect of bucketnumber on breastshot waterwheel performance." In *IOP Conference Series: Earth and Environmental Science*, vol. 105, no. 1, p. 012031. IOP Publishing, 2018.
[10] S. Darmawan, "Pengembangan model turbulen RNG k- ε untuk aplikasi CFD pada runner cross-flow dalam komponen turbin gas mikro bioenergy proto x-2a," Universitas Indonesia, Depok, 2015.
[11] Dendy Adanta, Budiarso, Warjito, A.I. Siswantara, "STUDY OF TURBULENCE MODELLING ON PICO-HYDRO TURBINE," in *Quality in research*, 2017.
[12] Streeter, Victor L., and E. Benjamin Wylie. "Mekanika fluida jilid 1." *Penerbit Erlangga, Jakarta* (1999).
[13] Schlichting, Hermann, Klaus Gersten, Egon Krause, Herbert Oertel, and Katherine Mayes. *Boundary-layer theory*. Vol. 7. New York: McGraw-hill, 1955.
[14] Munson, Bruce R., Donald F. Young, and Theodore H. Okiishi. "Mekanika Fluida Jilid 2." *Jakarta: Erlangga* (2005).
[15] Saad, Tony. "Turbulence modeling for beginners." *University of Tennessee space institute* (2011).
[16] Tennekes, Hendrik, and John Leask Lumley. *A first course in turbulence*. MIT press, 1972.
[17] Bakker, André. "Lecture 9-Kolmogorov's Theory Applied Computational Fluid Dynamics." *Educational material.© Fluent Inc* (2002).
[18] Chaudhry, M. Hanif. *Open-channel flow*. Springer Science & Business Media, 2007.
[19] Davidson, Lars. "Fluid mechanics, turbulent flow and turbulence modeling." *Div. of Fluid Dynamics, Dep. of Applied Mechanics, Chalmers University of Technology, Göteborg, Sweden* (2011).
[20] Rand, From Ingersoll. "Cameron hydraulic data: A Handy reference on the subject of hydraulics, and steam." (2002).
[21] Lesieur, Marcel. *Turbulence in fluids: stochastic and numerical modelling*. Boston, MA: Nijhoff, 1987.
[22] Menter, F. L. O. R. I. A. N. R. "Zonal two equation kw turbulence models for aerodynamic flows." In *23rd fluid dynamics, plasmadynamics, and lasers conference*, p. 2906. 1993.
[23] Menter, Florian R. "Improved Two-Equation k-εTurbulence Models for Aerodynamic Flows." *NASA Technical Memorandum* 103975 (1992).
[24] Aeschlimann, Vincent, Sébastien Beaulieu, Sébastien Houde, Gabriel Dan Ciocan, and Claire Deschênes. "Inter-blade flow analysis of a propeller turbine runner using stereoscopic PIV." *European Journal of Mechanics-B/Fluids* 42 (2013): 121-128.
[25] Faber, Tom E. *Fluid dynamics for physicists*. Cambridge University Press, 1995.
[26] Ramos, H. M., M. Simão, and A. Borga. "CFD and experimental study in the optimization of an energy converter for

low heads." *Energy Science and Technology* 4, no. 2 (2012): 69-84.

[27] Simpson, R. G., and A. A. Williams. "Application of computational fluid dynamics to the design of pico propeller turbines." In *Proceedings of the international conference on renewable energy for developing countries*. University of the District of Columbia Washington, DC, 2006.

[28] Prasad, Vishnu, V. K. Gahlot, and P. Krishnamachar. "CFD approach for design optimization and validation for axial flow hydraulic turbine." (2009).

[29] Ramos, Helena M., Mariana Simão, and A. Borga. "Experiments and CFD analyses for a new reaction microhydro propeller with five blades." *Journal of Energy Engineering* 139, no. 2 (2012): 109-117.

[30] Jeong, Jinhee, and Fazle Hussain. "On the identification of a vortex." *Journal of fluid mechanics* 285 (1995): 69-94.

[31] Gupta, Vishal, Vishnu Prasad, and Ruchi Khare. "Numerical simulation of six jet Pelton turbine model." *Energy* 104 (2016): 24-32.

[32] Veselý, J., and M. Varner. "A case study of upgrading of 62.5 MW Pelton turbine." In *Proceedings of the International Conference, IAHR*. 2001.

[33] Budiarso, Warjito, Dendy Adanta, N.S. Putra, H. Vohra, “Type Of Cut-Out Bucket Selection For Pico Hydro Pelton Turbine,” in *The 2nd International Conference on Engineering and Technology for Sustainable Development (ICET4SD)*, 2017.

[34] Correa, Jorge Luis Clarembaux, Jesús De Andrade, and Miguel Asuaje. "A Preliminary Analysis of a Turgo Type Turbine CFD Simulation Designed With an Integrated Dimensional Methodology." In *ASME 2012 fluids engineering division summer meeting collocated with the ASME 2012 heat transfer summer conference and the ASME 2012 10th international conference on nanochannels, microchannels, and minichannels*, pp. 327-337. American Society of Mechanical Engineers, 2012.

[35] Aaraj, Youssef, Sorina Mortada, Denis Clodic, and Maroun Nemer. "Design Of A Turgo Two-Phase Turbine Runner." (2014).

[36] Židonis, Audrius, David S. Benzon, and George A. Aggidis. "Development of hydro impulse turbines and new opportunities." *Renewable and Sustainable Energy Reviews*51 (2015): 1624-1635.

[37] Siswantara, Ahmad Indra, and Steven Darmawan. "Investigation of Inverse-Turbulent-Prandtl Number with Four RNG ke Turbulence Models on Compressor Discharge Pipe of Bioenergy Micro Gas Turbine." *Applied Mechanics & Materials*819 (2016).

[38] Sinagra, M., V. Sammartano, C. Aricò, and A. Collura. "Experimental and numerical analysis of a cross-flow turbine." *Journal of Hydraulic Engineering* 142, no. 1 (2015): 04015040.

[39] Kaniecki, Maciej. "Modernization of the outflow system of cross-flow turbines." *Task Quarterly* 6, no. 4 (2002): 601-608.

[40] Dragomirescu, Andrei, and Mihai Schiaua. "Experimental and numerical investigation of a Bánki turbine operating far away from design point." *Energy Procedia* 112 (2017): 43-50.

[41] Hunt, Julian CR, Alan A. Wray, and Parviz Moin. "Eddies, streams, and convergence zones in turbulent flows." (1988).

[42] Nakagawa, Hiroji, ed. *Turbulence in open channel flows*. Routledge, 2017.

[43] Quaranta, Emanuele, and Roberto Revelli. "Hydraulic behavior and performance of breastshot water wheels for different numbers of blades." *Journal of Hydraulic Engineering*143, no. 1 (2016): 04016072.

[44] Speziale, Charles G., and S. Thangam. "Analysis of an RNG based turbulence model for separated flows." *International Journal of Engineering Science* 30, no. 10 (1992): 1379-IN4.

[45] Steven Darmawan, Ahmad Indra Siswantara, Budiarso, Asyari Daryus, Agus Tri Gunawan, Achmad Bayu Wijayanto, Harto Tanujaya, “TURBULENT FLOW ANALYSIS IN AUXILIARY CROSS-FLOW RUNNER OF A PROTO X-3 BIOENERGY MICRO GAS TURBINE USING RNG k-ε TURBULENCE MODEL,” *ARPN Journal of Engineering and Applied Sciences* 10 (2015): 7086–7091.

[46] Toffolo, Andrea, Andrea Lazzaretto, and Antonio Dario Martegani. "An experimental investigation of the flow field pattern within the impeller of a cross-flow fan." *Experimental Thermal and Fluid Science* 29, no. 1 (2004): 53-64.

[47] Sun, Kangjie, Hua Ouyang, Jie Tian, Yadong Wu, and Zhaohui Du. "Experimental and numerical investigations on the eccentric vortex of the cross flow fan." *International Journal of Refrigeration* 50 (2015): 146-155.

[48] Nunes, R. A., and J. H. Simpson. "Axial convergence in a well-mixed estuary." *Estuarine, Coastal and Shelf Science* 20, no. 5 (1985): 637-649.

[49] Warjito, Budiarso, A. I. Siswantoro, Dendy Adanta, M. Kamal, and R. Dianofitra. "SIMPLE BUCKET CURVATURE FOR DESIGNING A LOW-HEAD TURGO TURBINE FOR PICO HYDRO APPLICATION." *INTERNATIONAL JOURNAL OF TECHNOLOGY* 8, no. 7 (2017): 1239-1247.

[50] Truesdell, Clifford. *The kinematics of vorticity*. Vol. 954. Bloomington: Indiana University Press, 1954.